DOI: http://dx.doi.org/10.12989/cac.2011.8.6.717

Nonlinear finite element analysis of RC beams strengthened with CFRP strip against shear

Nalan Bulut, Özgür Anil* and Çağatay M. Belgin

Department of Civil Engineering, Gazi University, Maltepe, Ankara, Turkey, 06570

(Received June 6, 2010, Revised February 25, 2011, Accepted March 9, 2011)

Abstract. Strengthening of reinforced concrete (RC) members against shear that is one of the failure modes especially avoided by using carbon fiber reinforced polymer (CFRP) is widely used technique, which is studied at many experimental studies. However, conducting experimental studies are required more financial resources and laboratory facilities. In addition, along with financial resources, more time is needed in order to carry out comprehensive experimental studies. For these reasons, a verified finite element model that is tested with previous experimental studies can be used for reaching generalized results and investigating parameters that are not studied. For this purpose, previous experimental study results are used and "T" cross-sectioned RC beams strengthened with CFRP strips with insufficient shear strength are modeled by using ANSYS software. First, finite elements modeling of the previously tested RC beams are done, and then the computed results are compared with the experimental ones whether they are matched or not. As a result, the finite element model is verified. Later, analyses of the cases without any test results are done by using the verified model. Optimum CFRP strip spacing is determined with this verified finite element model, and compared with the experimental findings.

Keywords: ANSYS; finite element analysis; RC beam; strengthening; nonlinear analysis.

1. Introduction

Technology emerges very rapidly nowadays with lots of projects that affects the human life. In these projects, safety is the most important factor. Besides this increase in labor force and raw material prices forces engineers to obtain optimum conditions. Modeling is defined as generating a structure in the computer aided environment and calculating the behaviors of it under defined loads. Thus study on this numerical model and obtaining the optimum conditions can be possible, in addition foreseeing the structural or material deficiencies are also possible. Modeling is used in engineering, medicine, transportation, aerospace, astronomy and in many other scientific fields. A structural member must be functional, safe and economic. Although these properties of structure are accomplished, there might be some faults at future. In order to prevent these future defects at the structure, it is good to know the deficiencies beforehand. This is the point, where the importance of modeling comes out.

Structural behavior under various loading conditions can be observed, when the structure is modeled in computer software. It is possible to determine the deficiencies in the structure and take necessary precautions by using the numerical modeling. By the help of these studies, the most economic cross sections and the structural element sizes can also be determined. Although approximate solutions are

* Corresponding author, Ph.D., E-mail: oanil@gazi.edu.tr

obtained from numerical models compared with the analytical or test results, it is faster and cheaper to conduct a study by using a numerical model than using an experimental study. In the scope of this study, beam specimens that are strengthened against shear by using CFRP (Carbon Fiber Reinforced Polymer) composites are modeled. The nonlinear behaviors of the reinforced concrete beam became even more complex, when CFRP strips are used for to strengthening. In order to model such a nonlinear beam behaviors as realistic as possible a finite element program which can consider nonlinear behaviors of materials is used. The name of this program is ANSYS.

CFRP composites are became widely used, popular material for repairing and strengthening processes due to its high tensile strength, easy of application and lightness. As the use of this material is increased the researchers studied more on the material experimentally and by modeling. It is well known that, experimental studies need much more financing, time and laboratory resources than the studies that are executed in digital medias. Testing limited number of specimens in the laboratory and than simulating the specimens in a software program became widely used technique. After that verified model is used for obtaining the behavior of the specimens rather carrying out a comprehensive experimental program, at which many parameters are tested at the same time. In the scope of this study, the above explained technique is applied to the T cross-sectioned reinforced concrete beams, which are strengthened with CFRP strips to obtain a relation between CFRP strip spacing and the load carrying capacities of the beams.

2. Literature survey

In this section some, of the important recent studies in the scope of this study are summarized.

Qu strengthened prestessed reinforced concrete specimens with CFRP panels and modeled them with ANSYS finite element program in his MSc thesis for comparing both results (1994). In the finite element modeling of the structures, he assumed perfect bonding between concrete-reinforcement and CFRP-concrete, and he did not consider aggregate interlocking. It is assumed that both cracking and crushing occurs in the concrete as the failure criterion. He stated that the successes in computing the load-displacement curve correctly depend on making correct assumptions for the stress-strain relationship, failure criterion and the amount of loss in the tensile strength due to cracks. It is found that the assumptions that are made for amount of loss in tension strength are closely fit with the assumption made in the load-displacement. Besides, in both theoretical and experimental studies the strength and the rigidity of the strengthened beams with CFRP are considerably higher than the control beams (Qu 1994).

Fanning (2001) both tested and modeled reinforced concrete post tensioned beams in his studies. He used ANSYS 5.5 finite element program for modeling. Solid65 and Link 8 element types are used for concrete and reinforcements, respectively. He used concrete element type that considers the non-linear behavior of concrete up to the failure in his model. Smeared crack model is used, which allows cracking of the concrete whether the reinforcement is modeled discrete or smeared. He states that the optimum strategy is to control the mesh density and discreet modeling of the reinforcement as a starting point. He concluded that, smeared crack is a reasonable model for modeling the reinforced concrete structures under pure bending (Fanning 2001).

Chansawat (2003) studied the nonlinear finite element analysis of reinforced concrete structures that are strengthened with FRP composites in his PhD. thesis. He dealt with the subject in three stages. Firstly he studied and evaluated the finite element model of the strengthened beams. Then secondly Horsetail Creek Bridge beams are casted, strengthened and results that are obtained from FEM and tests are compared. Finally the results of the beams before and after the strengthening with FRP under static and dynamic loadings with various load combinations are compared. He used nonlinear analysis to observe the structure's behavior accurately in his model. Displacements and shape changes are recorded and are compared with finite element analysis results. The concrete, reinforcements and FRP strips are modeled separately and perfect bonding is assumed between concrete-reinforcement and concrete-FRP. When the finite element model results are compared with experimental results, FEM is computed more slope for the load-displacement curve. This difference is obtained due to ignoring the micro cracking of the concrete in the model and the perfect bonding assumption according to Chansawat (Chansawat 2003).

Jia (2003) studied the nonlinear finite element analysis of reinforced concrete beams that are strengthened with FRP strips in his MSc. thesis. In his study, he strengthened the beams with CFRP strips by using organic and inorganic epoxies. Then he modeled the beams linear and nonlinear behavior, with finite element program and compared the modeled results with experimental ones. In the model Solid65, Link 8 and Solid 46 element types for concrete, reinforcements and CFRP strips are used, respectively. Perfect bond between reinforcements and concrete was assumed. Stiffnesses of finite element models are calculated more than the of test results in both linear and nonlinear analysis. This difference can be explained due to ignoring micro cracking in the model and making perfect bond assumption. The failure loads are computed between 5.4 to 25.9% more in finite element model due to ignoring the strain hardening behavior of concrete after yielding and taking assumed material properties into account instead of recorded values (Jia 2003).

Santhakumar, Chandrasekaran and Dhanaraj (2004) modeled the experimental research of Norris *et al.* (1997) by using ANSYS FEM program. In the model Solid65, Link 8, Solid 46 and Solid 45 element types for concrete, reinforcements, CFRP strips and supports with loading plates are used, respectively. It is assumed that there is a perfect bonding between concrete reinforcement and concrete CFRP. It is seen that there is less rotation at ultimate capacity load carrying at all of the numerical models than the experimental results. Higher values of stiffnesses are recorded at linear model than the test results (Santhakumar *et al.* 2004).

Elyasian, Abdoli and Ronagh (2006) modeled the FRP (Fiber Reinforced Polymer) strengthened beams at ANSYS finite element program. In the study, parameters such as fiber angle, concrete compressive strength, cross sectional areas of tension and compression reinforcements, amount of stirrup and spacing are researched. During modeling Solid65, Link8 and Solid45 element types for concrete, reinforcements and FRP strips are used, respectively. In addition Shell43 elements are used for the supports and loading plates. It is assumed that no slip is occurred between concrete reinforcement interface and between concrete FRP interfaces. In the study, ANSYS analyses are gave close results to test results. Thus the effect of FRP strip angle on the load carrying is proved (Elyasian *et al.* 2006).

Camata, Spacone, Zarnic (2007) the brittle failure modes of reinforced concrete members strengthened with FRP strips against bending at experimental and analytical studies investigated. In the analysis, smeared crack model is used in order to model the stiffness at post cracking and discrete crack model is used in order to compute the transient displacements at the nodes, when the concrete cover is separated. The effects of concrete cracks, plate length and width, stiffness and bonded layer behavior on the failure mechanism are investigated. They found that the failure started from the end of the short FRP plates and from the center of the span length in the long FRP plates. In addition, over strengthening with FRP against bending increases the load carrying capacity but decrease the ductility. Besides, finite element analysis showed that the distance between bending and shear cracks is a considerable parameter (Camata *et al.* 2007).

Hashemi, Rahgosar Maghsoudi (2007) investigated the usability conditions of the high load capacity beams that were strengthened with FRP plates in their studies. They handled the subject in four steps; firstly they tested high load capacity beams under high compressive strength that are strengthened with different arrangements of diagonally inclined FRP and reinforced with different tension reinforcements. Secondly the effects of application of different FRP plates on the bending strength are evaluated and then displacements and crack widths for assessing the usability conditions are done. Finally a three dimensional nonlinear finite element model is developed. In the model, Solid65, Link8 and Solid46 element types for concrete, reinforcements and FRP strips are used, respectively. It was assumed that there is a perfect bonding between the materials, and there is no debonding between concrete-reinforcement and concrete-FRP. The non linear behaviors of materials due to creep, shrinkage and heat variations are ignored. Concrete is considered as isotropic and orthotropic before cracking and after cracking, respectively. The reinforcement is assumed as isotropic. FRP material is considered as orthotropic along the fiber direction and isotropic along the width. As a result it can be modeled in two perpendicular directions. The load increments are made step by step in order to avoid the divergence at the nonlinear analysis of the beams. Furthermore the experimental results show that the crack widths are lesser in all load levels at high strength concrete (Hashemi *et al.* 2007).

In this study, a finite element model is developed which is verified by the experimental data obtained from the study of T cross-sectioned reinforced concrete beams strengthened against shear (Anil 2008) beside the studies mentioned above. A relation between CFRP strip width and reinforced concrete beam load carrying capacity is proposed for the beams by using the finite element model, which is verified by the experimental results.

2. The experimental study that forms a foundation for the finite element model

A total of 5 T-shaped beams are tested under reversed cyclic loading in the experimental program. The geometrical details and reinforcement schemes of the test specimens are given in Fig. 1. The cross sections and the longitudinal reinforcements of all of the specimens are identical.

The properties of the test specimens are summarized in Table 1. Mean standard cylinder compressive strength values of the concrete of the specimens are determined. All of the compressive strengths of the specimens are below 15 MPa as seen in Table 1. The test specimens are manufactured from low concrete compression strength on purpose.

Beam-1 is a control specimen without strengthening. The beams without shear reinforcement are strengthened with CFRP strips that are applied along to the shear span. Beam-2 and Beam-3 specimens are strengthened with CFRP strips and Beam-4 and Beam-5 specimens are strengthened with CFRP plates with different arrangements. The strengthening details of the specimens are given in Fig. 2. The experimental setup and the schematic view of the measurement equipments are given in Fig. 3. Test specimens are fixed to the rigid wall in the laboratory by using high strength steel bolts with a diameter of 45 mm. Reversed cyclic loading is applied to the specimens with a loading column that is hinged at both ends mounted between the rigid slab and the specimen. The ratio of shear span length (a = 1675 mm) to the effective height (d = 335 mm) of all specimens is a/d = 5.0. The displacements at the end points of the beams and the curvature along the maximum moment

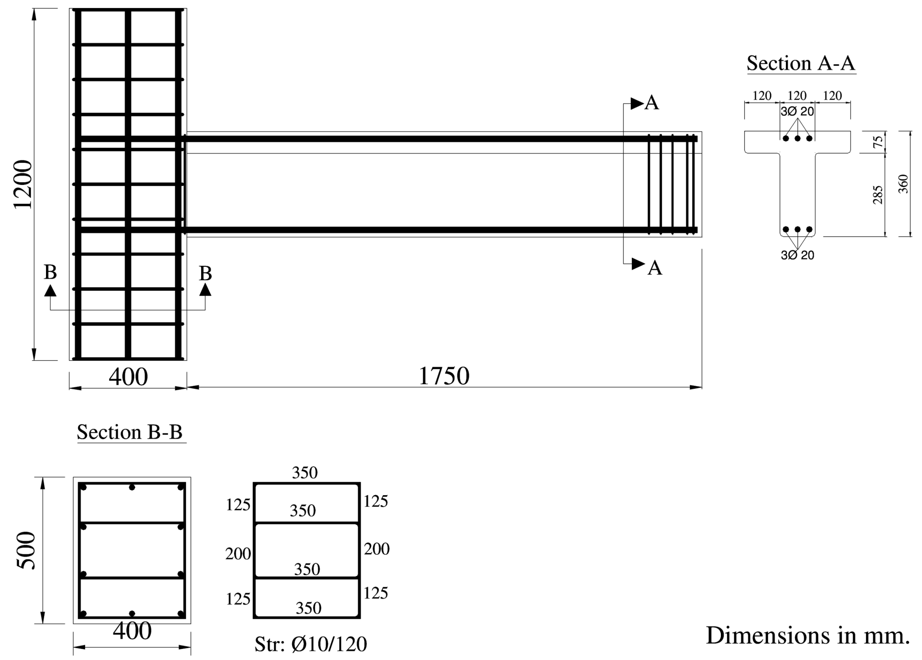


Fig. 1 Test specimens for the experimental study (Anil 2008)

Table 1 Properties of test specimens

Test specimen #	*a/d*	f_c (MPa)	Properties of CFRP strips		
			w_f Width (mm)	s_f* Spacing (mm)	Bonding form
Beam-1	5.0	15.0	-	-	-
Beam-2	5.0	14.0	50	80	U type
Beam-3	5.0	14.5	100	130	U type
Beam-4	5.0	14.8	1750	-	Only two side faces
Beam-5	5.0	14.2	1750	-	U type

*The distance between two consecutive CFRP axes. (Ref. Fig. 2)

region are all recorded by using LVDT's. In addition strain on the CFRP strips are recorded electronically. The strains are measured from the CFRP along the shear span of the beam that is bonded 300-1000 mm away from the rigid wall. Strain is measured along the fiber direction of the CFRP. The load-displacement behavior depends on to the details of the strengthening. The load-displacement envelopes of the specimens are given in Fig. 4. Detailed information about the experiment results is presented at reference Anil (2008).

3. ANSYS finite element model

Finite element program ANSYS is chosen for modeling the structure. First ANSYS finite element

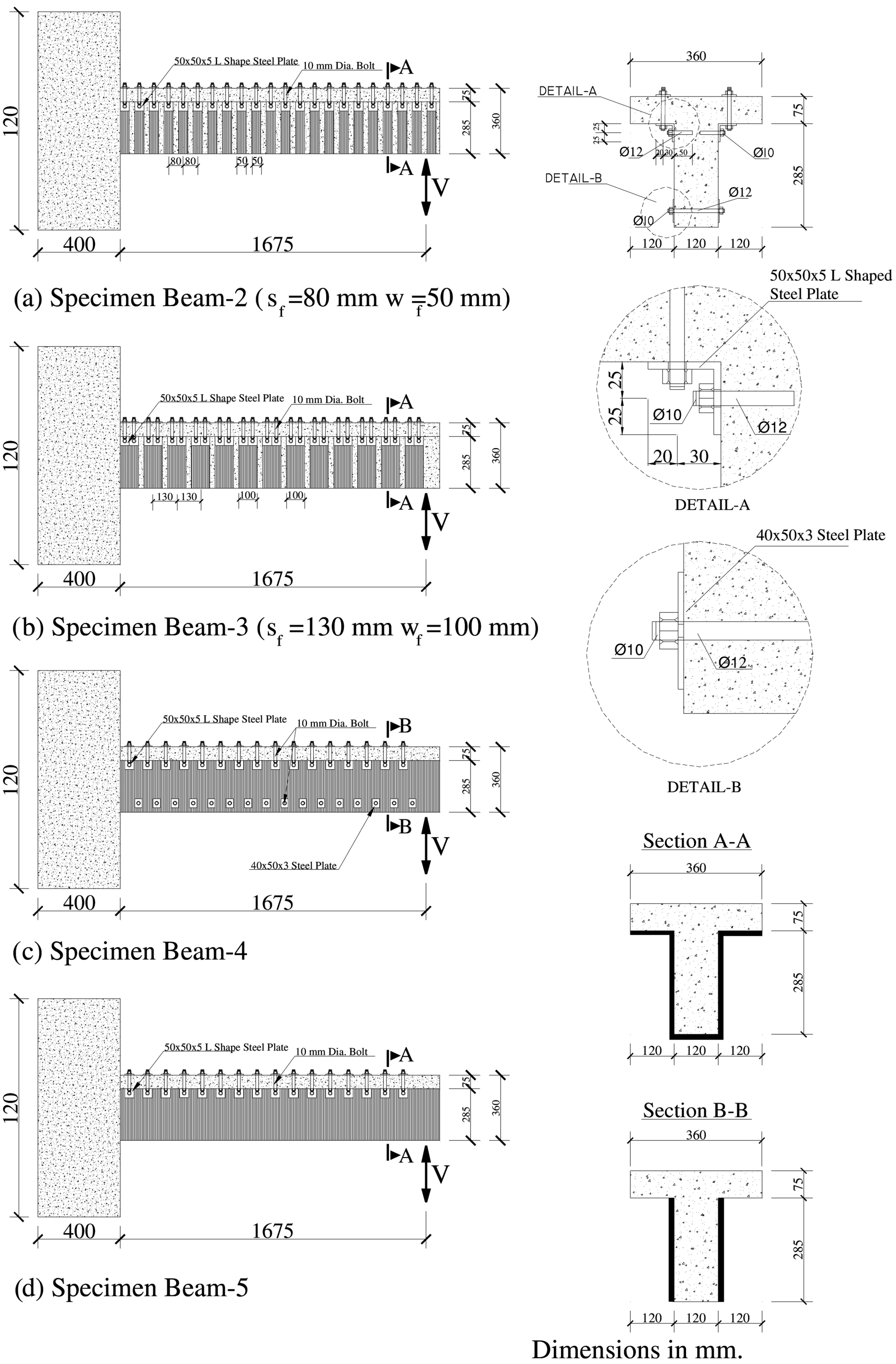
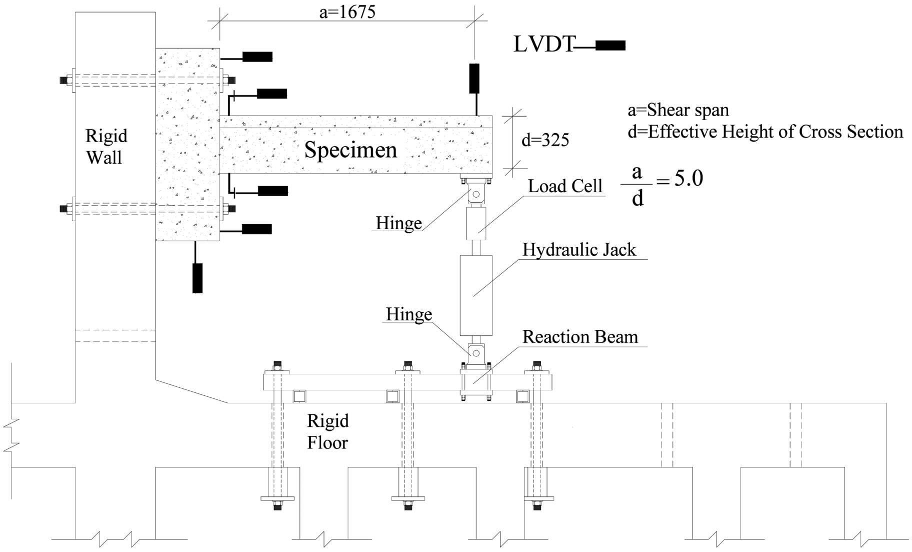


Fig. 2 The strengthening details applied to the specimens (Anil 2008)

program is developed by Swanson Analysis System Company at 1971. Till then the program is modified many times up to nowadays. The program is widely used and approved for engineering



Dimensions in mm.

Fig. 3 Test setup and instrumentation (Anil 2008)

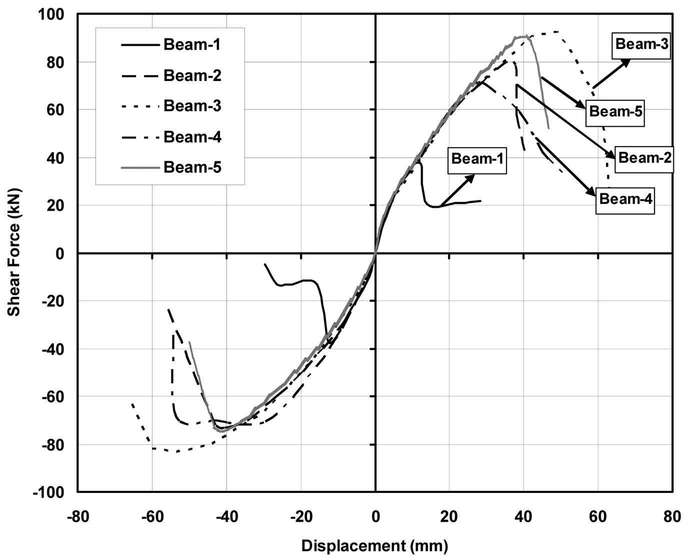


Fig. 4 The load-displacement envelopes of the specimens (Anil 2008)

applications and academic studies. ANSYS use many different element types that can compute nonlinear behaviors of materials. An element named “Solid65” is used for modeling the nonlinear behavior of the concrete.

Due to those features, ANSYS finite element software is chosen for modeling the beams that are strengthened with CFRP in this study. The analysis steps that are used during the modeling at ANSYS Finite Element Software are given in Fig. 5 and are explained in this section of the study.

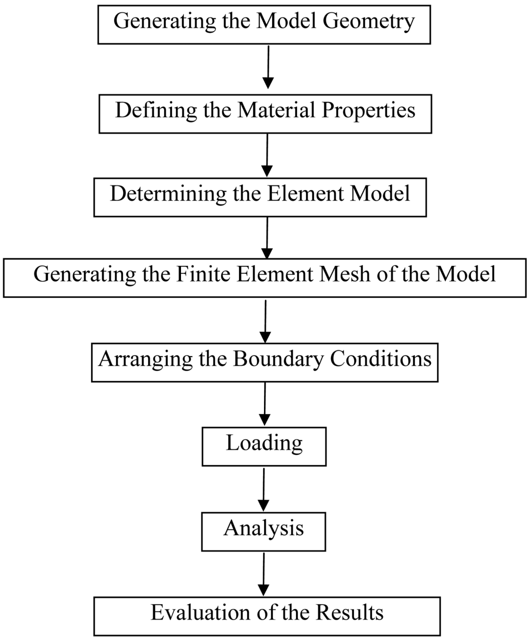


Fig. 5 Analysis steps of ANSYS

3.1 Generating the model geometry

Beams are modeled with volumes while generating the volumes, the location of the reinforcements is taken into account. Because they are assumed to be scattered into the concrete. Furthermore the concrete cover is also taken into account during the modeling of the volumes. That is the reason, why the model of the beam is consisted of many volumes that came together instead of a single volume. The advantage of the symmetry of the beams is used to model the half of the span length of the beams. The volumes that are used at the model are given at Fig. 6 for Beam-3 as a sample. Concrete with reinforcements, concrete without reinforcement and CFRP strips are modeled separately in the model. This modeling technique provides ease of selection of the volumes

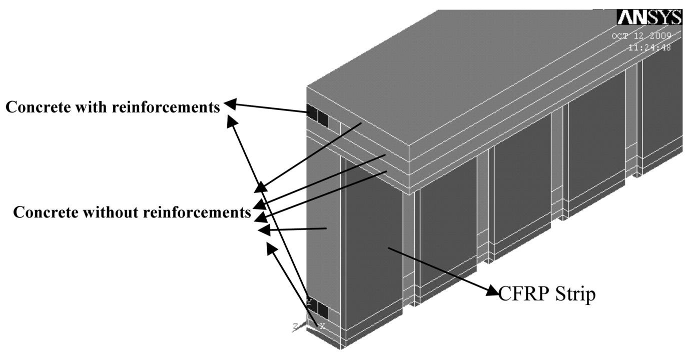


Fig. 6 Beam-3: model volumes used for the specimen

separately, as a result finite element mesh can be generated easily.

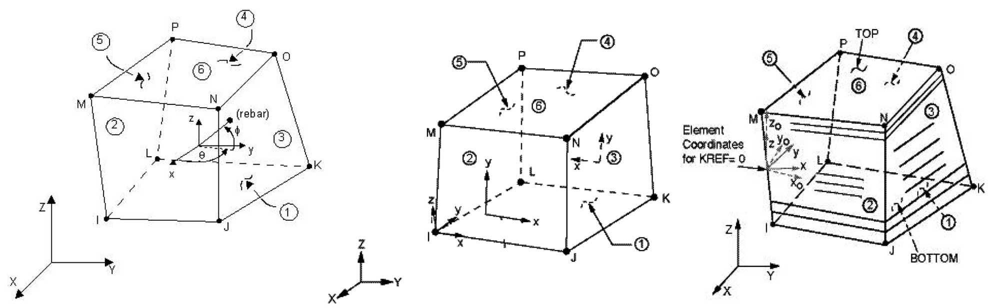
3.2 Defining the material properties

The successful analyses of the problems at numerical methods rely on the successful definition of the material properties as close as the original behavior of that material. In this study the material properties of concrete, reinforcements and the CFRP are defined separately. The well recognized Hognestad unconfined concrete model is used to define the concrete (Ersoy and Özcebe 2010). Hognestad model is used to define the behavior of the concrete under compression loads. The multilinear isotropic hardening option is chosen for the concrete, and stress-strain values that are obtained from the Hognestad model are entered into ANSYS. The failure criterion of the concrete is defined by William-Warnke model (William and Warnke 1974). This model is defined with nine constants that are entered into ANSYS software. During modeling of the reinforcements' linear elastic material model is used and elastic modulus, yield strength and poisson ratio values are entered. High tension capacity is the most important characteristic behavior of the CFRP. However CFRP is an unisotropic material, the directions of the fibers affect the behavior significantly. Linear elastic material model is used to define the CFRP in this study.

3.3 Determining the element model

Solid65 element type is used for modeling concrete at ANSYS software. This element type is used for modeling concrete at almost all of the previous studies. In this study, smeared modeling is used for the reinforcements. No other element type is used for modeling the reinforcements. Solid46 element type is used for modeling the CFRP. Load is not applied on to a single point, but instead a loading plate is modeled at the point where the loading is applied in order to avoid the stress concentrations and the local cracks. Solid45 element type is used for this plate. Finite element types that are used for the model are given in Fig. 7.

Solid65 element type is an 8-node element, which is used in three dimensional modeling of concrete that considers cracking in tension and crushing in compression. This element had three directional degree of freedom at each node (Fig. 7(a)). Both concrete and reinforced concrete can be modeled with this element. The most important feature of this element type is the nonlinear material modeling capability. Thus the cracking, crushing, plastic deformation and creep behaviors of concrete and reinforcement in three orthogonal directions can be modeled under compression and



(a) Concrete (Solid65) (b) Steel plate (Solid45) (c) CFRP (Solid46)

Fig. 7 Element types used for the finite element model tension. Solid45 element type has eight nodes with three degrees of freedom at each node along the three directions and can be used for computation of creep, shrinkage, high values of displacements and rotations (Fig. 7(b)). Solid46 is a three dimensional eight node element with three degree of freedom at each node (Fig. 7(c)). It is used for modeling the thick shells and solids. A solid member with 250 different types of layers can be modeled with the help of this element easily. This element type is used for modeling the CFRP strips for this study and the CFRP strips are considered as a single layered structure. Orthotropic material model can be used with this element.

3.4 Generation of finite element mesh

Finite element method can be defined briefly as dividing the structure into small geometrical elements and creating a finite element mesh according to the geometrical shape of the structure. Determining the mesh density is one of the most important processes of generation of a mesh. Another important point is

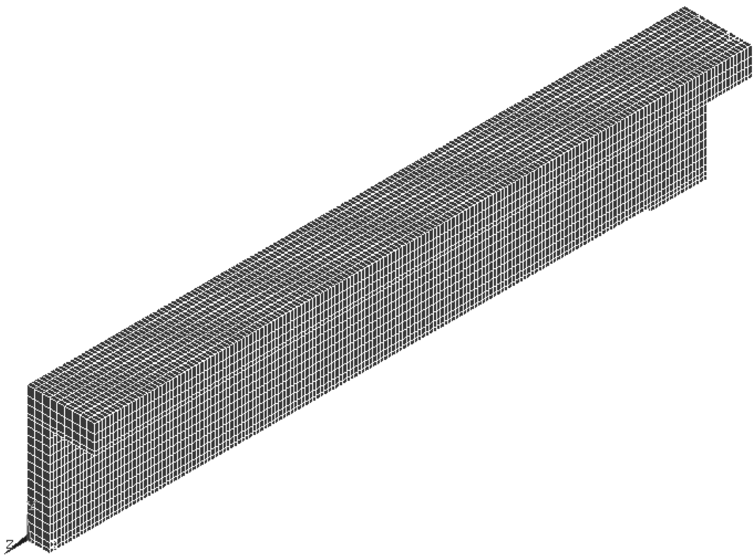
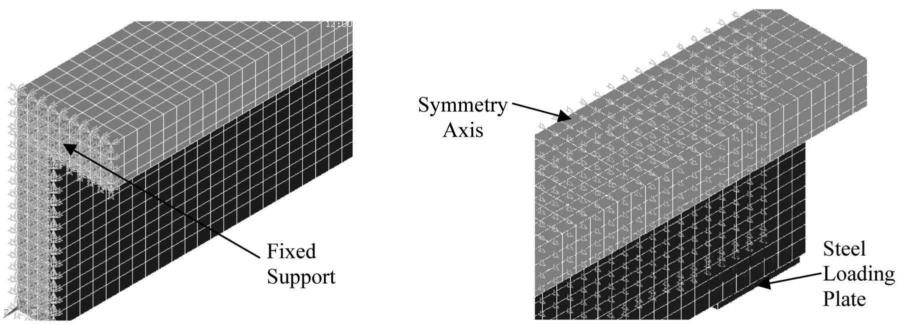


Fig. 8 Finite element mesh of Beam-1



(a) Boundary conditions in fixed end (b) Boundary conditions in symmetry axis

Fig. 9 Boundary conditions of the finite element model that the geometries are meshed with consisted sized elements. The size differences kept small (Moaveni 1999, Lawrence 2002). Beam-1 finite element mesh is given in Fig. 8 as a sample.

3.5 Arranging the boundary conditions

During the tests, the beams are fixed to a rigid wall at one end and are loaded from the free end. While finite element model is prepared displacements and the rotations of all orthogonal directions of the nodes at the fixed end of the beam are kept. Due to geometrical symmetry, half of the beams are modeled. In order to define the symmetry axis of the model, the displacements in x-axis of the nodes on the symmetry axis are kept and only the displacements in y-axis are set free (Fig. 9).

3.6 Loading

The load is applied to the free end of the test specimens by a hydraulic jack. The finite element model is prepared identical with the test setup including the loading plate. Load is applied by a steel plate, which is modeled with Solid45 elements (Fig. 9(b)).

3.7 Analysis

Nonlinear analysis method is chosen for the finite element analyses of the previously tested reinforced concrete beams that are strengthened with CFRP strips. The main reasons for choosing this method is that the reinforced concrete beam responds nonlinearly due to material characteristic under loading and that the loading is applied step by step. There are some parameters like load step, sub step and time at ANSYS software, when a nonlinear analysis are carried out. Related with the complexity of the model these values are quite high in order to have a better observation of the behavior of the material and the yield point or the elasticity limit on the load-displacement curve (Fig. 10).

Newton-Raphson method is used for non-linear analysis at ANSYS. The applied load is divided in to loading steps in this method. These load steps are also divided into sub steps. Analysis last until convergence condition is satisfied. Convergence criterion in each solution is determined

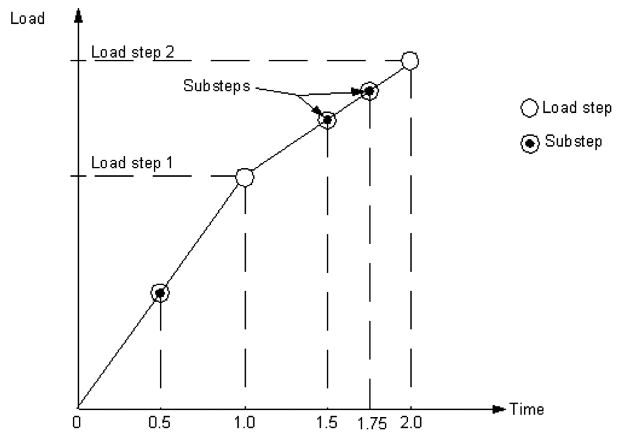


Fig. 10 Nonlinear analysis steps of ANSYS (Moaveni 1999, Lawrence 2002)

according to the unbalanced load vector. The unbalanced load vector is the difference between the applied load and the sum of the element stresses. The program uses this vector for solution. In a divergent case this vector and the rigidity matrix is recalculated for a divergent case and a new solution is obtained (Moaveni 1999, Lawrence 2002).

4. Comparison of the finite element analysis with the experimental results

Before making the comparison of experimental and finite element analyses results that are made for strengthened T shaped reinforced concrete beams with low compression strength by using CFRP strips against shear, the assumption that are made are summarized below.

While modeling the concrete, its elastic and plastic behavior with its failure behavior is considered. Reinforcement is modeled with only considering its yielding and ruptures stress values. CFRP is modeled like a linear-elastic material. Solid65 element type is used for concrete. Due to difficulties determining the contact surfaces under the loading and additional requirement of processing power of this process, reinforcements are modeled with smeared model (considered as scattered) instead of using discrete elements for the reinforcement. Solid46 and Solid 45 element types for CFRP strips and loading plate is used, respectively. It is assumed that there is no debonding, and perfect bonding between concrete-reinforcement and concrete-FRP is obtained. Only half of the beams are modeled due to geometrical symmetries of the beams. Mesh density is one of the most important factors in finite element methods for accurate results. Hence in order to determine the appropriate mesh density, two different element sizes of 10 mm and 25 mm are used for generating meshes. Between 5600 and 18000 elements are generated in the strengthened models considering the various CFRP arrangements.

The generated ANSYS model for this study is verified by the comparison of the load-displacement curves of both the test results and the computed modeling results of the first five beams. In order to obtain a relation between CFRP strip spacing and beam load carrying capacities, two beams without experimentally tested are analyzed, and the load-displacement curves and the load carrying capacities are obtained with verified model. The properties of the beams that are not tested but only modeled in ANSYS are given in Table 2.

The selected examples of load-displacement comparison envelopes of the finite element analysis results and the experimental results are given in Fig. 11. It can be seen from figure that the finite element analysis and experiment shear force results are considerably close to each other. Initial stiffnesses are calculated close to that of experimental results. However failure displacement values of finite element analysis are calculated less than that of the experimental ones. Load carrying capacities obtained from the tests and analysis are given in Table 3.

Table 2 Modeled in ANSYS but not tested beam properties

Model #	a/d	f_c (MPa)	Properties of CFRP strips		
			w_f Width (mm)	s_f* Spacing (mm)	Bonding form
Beam-6	5.0	15.0	50	120	U type
Beam-7	5.0	15.0	50	150	U type

*The distance between two consecutive CFRP axes.

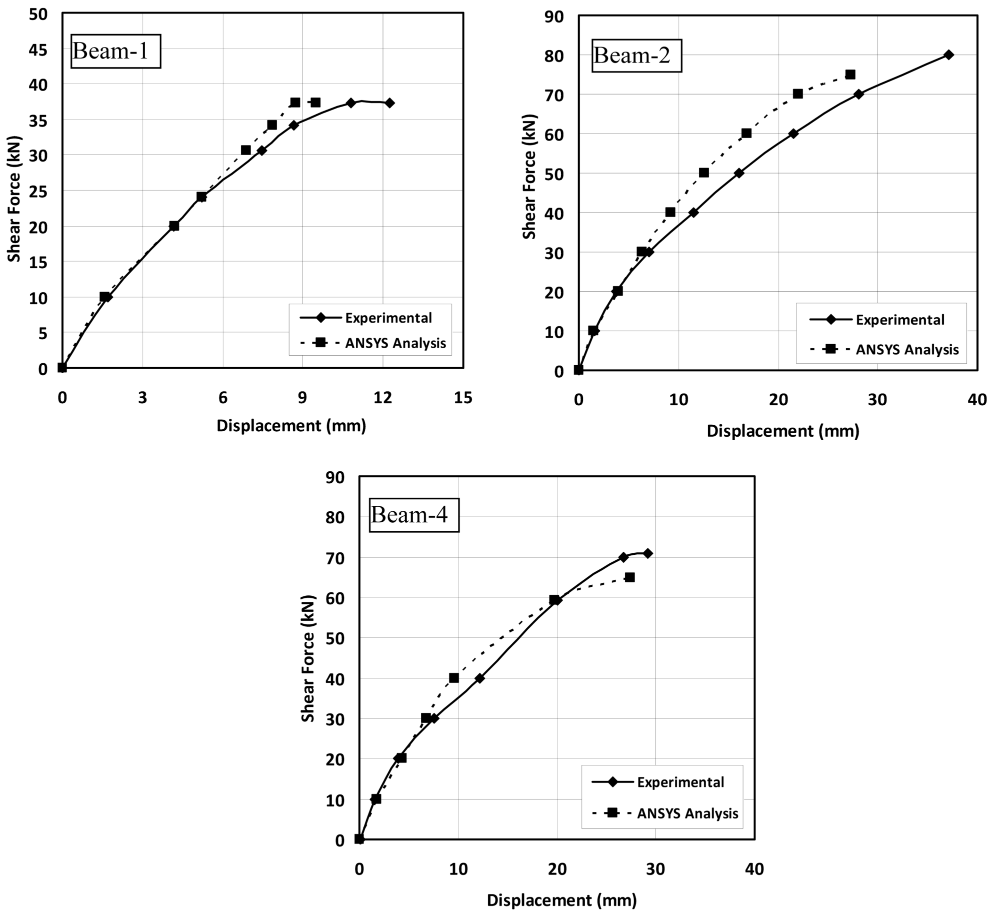


Fig. 11 Comparison of experimental and analytical load-displacement graphs

The average load carrying capacities of the models are computed 5.5% less than the experimental results for the first five analyses that are made for verifying the finite element model. The analysis results for all five specimens are lower than the test results and they are at the safe side. The authors thought that both analyses and test ultimate load carrying capacities are very close to each other with in an acceptable range.

The difference between the analysis results and the experimental results for the failure load displacement increased when compared with respect to load carrying capacity values. The average experimental failure displacement values are 27% higher than the analysis results. The authors thought that the difference is due to the crack model used in ANSYS and due to the perfect bonding assumption between CFRP strips and the concrete instead of using another contact element in between. Similar results are encountered in the previous studies. In finite element method as Lu, Jiang, Teng, Ye (2006) stated in their studies, there are two methods for modeling the materials that are bonded to each other. In one method, interface elements are used and in the other method without using any interface elements, crack models of bonded materials are used for defining the failure of each material. In most of the commercial finite element analysis programs the second method is used for defining the failure of the materials. Finite element programs like ANSYS, ABAQUS uses the fixed

Table 3 Comparison of the experimental and the finite element analysis capacities

Beam No	FEM analysis (kN)	Experimental failure load (kN)	Difference (%)
Beam-1	37.3	37.3	0
Beam-2	75.0	80.0	6
Beam-3	82.2	92.3	12
Beam-4	64.8	70.8	8
Beam-5	88.0	89.3	1
Beam-6	72.1	-	-
Beam-7	61.2	-	-

FEM: Finite element model

angle crack model (FACM) for the concrete. In the FACM model the crack occurs in a perpendicular direction to the principle stresses and proceeds without changing its direction. FACM is incapable for modeling the shear softening and assumes an increase in the shear stress distribution in post cracking zone. Hence the researchers study different crack models for modifying the existent model (Lu *et al.* 2006). In addition to the crack models a special type of contact elements with user defined mechanical properties can be used between concrete and CFRP. There are such contact elements in ANSYS element library.

The authors thought that the finite element model generated with ANSYS software is quite successful for calculating the load carrying capacities of the reinforced concrete beams that are strengthened with CFRP strips considering the first five analyses in this study. The load carrying capacities of the beams are computed with the finite element model in the proximity of the experimental results. The error is very small and at the safe side. Hence two more analysis is carried out with the verified finite element model for determining a relation between the CFRP strip spacing and the load carrying capacity. In

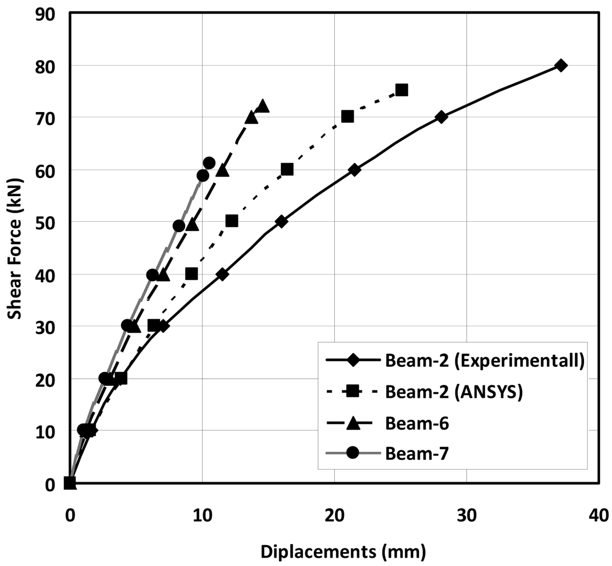


Fig. 12 Load-displacement curves of the beams that are not tested but modeled with finite element method

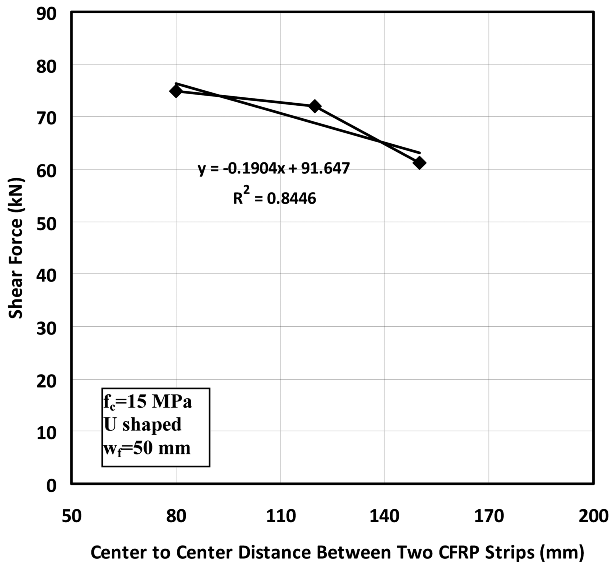


Fig. 13 The relation between CFRP strip spacing and load carrying capacity for 50 mm CFRP strip width

these two analyses, the beams are not tested but only modeled. The load displacement figures obtained from analysis results are given in Fig. 12 with the test results with similar geometrical conditions for comparison. Load carrying capacities are given in Table 3.

In the analyzed beams the concrete compressive strength (15 MPa), shape of the CFRP application (U shaped) and CFRP strip width (50 mm) are kept constant. Only variable is the CFRP spacing. The load carrying capacities of the T cross sectioned and strengthened beams as mentioned above are computed and given in Fig. 13 for various strip spacing. However it should not be forgotten that the graph, which is given in Fig. 13, is only suitable for the beams with properties that are described above. The properties that are considered as constants are chosen among the frequently used ones for most of the applications.

It could be seen from the Fig. 13 that the variation between CFRP strip spacing and the load carrying capacity is almost a linear line. Load carrying capacity decreases with the increase in CFRP strip spacing. This decrease is taken to be linear. The equation of the fitted line is also given in Fig. 13. It could also be seen from the Fig. 13 that the R^2 of the line is 0.85 that is close to the finite element analysis results. The approximate load carrying capacity of a beam can be calculated for different strip spacing with the help of this equation.

5. Conclusions

In the scope of this study T-shaped reinforced concrete beams with low compressive strength that are strengthened against shear with CFRP strips are modeled in ANSYS finite element program. Different material modeling capabilities of ANSYS for concrete and reinforcements are applied to model for computing and comparing the load-displacement curves. Although the failure loads that are obtained from analysis are very close to the experimental results for all beams, there are some differences at displacement values.

The authors thought that a 5.5% difference between finite element and experimental results at failure loads of the beams can be considered as acceptable in finite element analysis. Two more beams identical with Beam 2 CFRP strip widths but with different spacing are modeled and failure loads are computed with the help of verified ANSYS model. The model results show that the increase in CFRP spacing causes a decrease in the load carrying capacity of the beams.

In this study, although the failure loads are calculated successfully, the displacement can not calculated with similar success. This subject is mentioned also in the previous studies and is a consequence of the ANSYS crack model (FACM), which is not sufficient to model the concrete behavior (Lu *et al.* 2006).

Concrete is a nonlinear material with many micro cracks in its natural form. When the load is increased, those micro cracks join together and form a major crack zone thus causing a reduction in the load carrying capacity of the concrete and causing more nonlinear volume as a ratio in the concrete (Belgin and Şener 2008). When the analysis results of the Beam-1 (control beam) is compared to its experimental results, it can be said that the load-displacement and failure load results obtained from ANSYS are in harmony. Because the CFRP strip strengthening is applied to the Beam 2, 3, 4, 5, 6 and 7, therefore the nonlinear behavior of the concrete became more complicated resulting worse correlation between the experimental data and the finite element analysis results for displacement values. Also the unsymmetrical loading and the supports brought more complexity into this nonlinear behavior.

During the analysis, it is found that the change in the amount of the load increase had effect on the beam displacements. The displacements of the beams are computed more, when the load increments are kept low and the displacements of the beams are computed less with higher load increments. Thus for some beams number of load step is raised up to 65 to obtain more harmonic load-displacement curves with the test results. However during the analysis, the software created many subs files and stored many gigabytes of data. Hence the software needed a lot of free space in hard disk. When the numbers of the elements are about 18000, the file sizes increased up to 1000 gigabytes. The increase in the number of load step caused use of vast amount of the memory in the computer. Hence the number of load step should not be increased too much and should be kept in an optimum range unless it is necessary.

In this study, it is found that the failure loads of all beams are very close to that of experimental failure loads but the displacement values are not. In the future studies a better crack model needed for calculating the displacements in a better harmony with the test results. The authors thought that additional contact elements between CFRP strips and concrete surface gave more successful results for the load-displacement behavior. Besides in order to determine the effect of CFRP spacing on the load carrying capacity more models with various spacing and arrangements should be analyzed.

References

Anil, Ö. (2008), "Strengthening of RC T-section beams with low strength concrete using CFRP composites subjected to cyclic load", *Constr. Build. Mater.,* **22**, 2355-2368

Belgin, Ç.M. and Şener, S. (2008), "Size effect on failure of over reinforced concrete beams", *Sci. Eng. Fract. Mech.*, **75**, 2308-2319.

Camata, G., Spacone, E. and Zarnic, R. (2007), "Experimental and nonlinear finite element studies of RC beams strengthened with FRP plates", *Compos. Part B - Eng.*, **38**, 277-288.

Chansawat, K. (2003), "Nonlinear finite element analysis of reinforced concrete structures strengthened with

FRP laminates", Ph. D. Thesis, Oregon State University.
Elyasian, I., Abdoli, N. and Ronagh, H.R. (2006), "Evaluation of parameters effective in FRP shear strengthening of RC beams using FE method", *Asian J. Civil Eng. (Build. Housing)*, **7**(3), 249-257.
Ersoy, U. and Özcebe, G. (2001), *Reinforced concrete*, Evrim Publishing, İstanbul. (in Turkish)
Fanning, P. (2001), "Nonlinear models of reinforced and post-tensioned concrete beams", *Int. Electron. J. Struct. Eng.*, **2**, 111-119.
Hashemi, S.H., Rahgozar, R. and Maghsoudi, A.A. (2007), "Finite element and experimental serviceability analysis of HSC beams strengthened with FRP sheets", *Am. J. Appl. Sci.*, **4**(9), 725-735.
Jia, M. (2003), "Finite element modeling of reinforced concrete beams strengthened with FRP composite sheets", Master of Science Thesis, The University of Alabama in Huntsville.
Lawrence, K.L. (2002), *ANSYS tutorial release 7.0 and 6.1, SDC publications*, Canonsburg, 1.1-2.25.
Lu, X.Z., Jiang, J.J., Teng, J.G. and Ye, L.P. (2006), "Finite element simulation of debonding in FRP-to-concrete bonded joints, elsevier", *Constr. Build. Mater.*, **20**, 412-424.
Moaveni, S. (1999), *Finite element analysis, theory and applications with ANSYS*, Bill Stenquist, Prentice Hall Inc., New Jersey.
Santhakumar, R., Chandrasekaran, E. and Dhanaraj, R. (2004), "Analysis of retrofitted reinforced concrete shear beams using carbon fiber composites", *Int. Electron. J. Struct. Eng.*, **4**, 66-74.
Qu, R. (1994), "Theoretical analysis of reinforced and prestressed concrete bridge members strengthened with CFRP laminates", Master of Science Thesis, Florida Atlantic University.
William, K.J. and Warnke, E.P. (1974), "Constitutive model for the triaxial behavior of concrete", IABSE, Report No. 19, Bergamo, 1-30.

CC

Notations

a : Beam shear span length (mm)
d : Effective height of beam (mm)
f_c : Concrete compressive strength (MPa)
s_f : CFRP spacing length-from center to center (mm)
w_f : CFRP strip width (mm)